



Boardlinx Symbol Generator

Linking your FPGA to your circuit board

View us online:
<http://www.oledatech.com>

Boardlinx is a web-based design tool that offers hardware designers a quick path to generating board level schematic symbols that are customized to your specifications and design information. This tool can take the output from your FPGA place and route tools, or read your user constraint files so you can run your FPGA and board-layout design flows in parallel. Boardlinx will generate schematic symbols compatible with Orcad and Viewlogic board-level schematic flows. Additionally, Boardlinx generates graphical and text-based documentation to ease the transition from schematic entry to board layout and assembly test. Boardlinx also assembles documentation links specific to your part selection.

Today, larger FPGAs have over 1700 pins. Creating symbols by manually entering pin information is time-consuming and error prone. Simple typing errors and omissions can lead to costly debug time and board spins. Existing import functions and scripts written for CPLD and small FPGAs are cumbersome and inflexible. Boardlinx supports the latest tools and parts while providing the user with extensive configuration options. Creating schematic symbols can now be accomplished in **four easy steps**:



1. Read the design signal assignments
2. Set your configuration options
3. Adjust your pin spacing
4. Generate and download your symbols

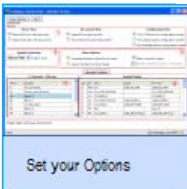
Reading your design information

If your FPGA design is complete, you can create symbols from the place and route report files such as "design.pad". If you can't hold up your board design, Boardlinx can read the pin assignments from the same User Constraints File that the Xilinx place and route tools use. You can also create a comma-separated-value file in text editor or Excel, which has the signal names and the pin locations. Once the part choice is specified, Boardlinx supplies the dedicated pins such as power, ground and JTAG required to complete your design package.

Contact Oleda
for a trial
account

Symbol Configuration Options

- Support for multiple parts with a common footprint
- Partitioning of design into multiple symbols
- Placement of VCCO, no-connect, and configuration pins
- Pin label options
- Partitioning of MGTs



Graphical Editor

The algorithm used by Boardlinx provides a starting point to fine tune your symbols. Signals declared as INPUT are placed on the left, the rest are placed on the right and busses are placed in order. Boardlinx also provides a graphical editor that allows you to reposition pins on a symbol or move pins to another symbol. Pins and spaces can be positioned to your standards.



Downloading your symbols

Boardlinx generates heterogeneous symbol sets for both Orcad and Viewlogic based design flows. Viewlogic symbols are generated directly and packaged in a zip file for easy download. Orcad symbols are described in EDIF format that can be imported to create an OLB library file.

Additional Documentation

Boardlinx also generates an interactive HTML based graphical representation of the design package. Assigned pins are readily identified and individual signals and busses can be highlighted in different colors. This tool provides a easy tool for planning and checking your pinout and board signal flow.



Boardlinx generates a custom pinout table that lists all pins, their dedicated functions and signal assignments. This table can be sorted by clicking on the headings. The table can easily be cut and pasted into desktop applications such as Word.



Links to the manufacturer's datasheets, package specifications, as well as application notes specific to the part are generated and available for the user.

Learn More about Boardlinx

Boardlinx is available as a subscription based design tool. All access to the tool is SSL encrypted, the same technology used for credit card transactions across the web. [Contact Oleda Technologies](http://www.oledatech.com) to set up a demonstration and trial account.